

ANALYSING THE CAVITATION PHENOMENA IN IN-NOZZLE FLOWS

Blaž Vajda

*University of Maribor, Faculty of mechanical engineering
Engine research Laboratory
Smetanova Street 17, SI-2000 Maribor, Slovenia
tel.: +386 2 2207740, fax: +386 2 2207990
e-mail: blaz.vajda@uni-mb.si*

Zoran Žunič

*AVL-AST d.o.o.,
Trg Leona Štuklja 5, SI-2000 Maribor, Slovenia
tel.: +386 2 2294805
e-mail: zoran.zunic@avl.com*

Breda Kegl

*University of Maribor, Faculty of Mechanical Engineering
Engine research Laboratory
Smetanova Street 17, SI-2000 Maribor, Slovenia
tel.: +386 2 2207732, fax: +386 2 2207990
e-mail: breda.kegl@uni-mb.si*

Abstract

The geometry of the diesel fuel injection nozzle and fuel flow characteristics in the nozzle significantly affects the processes of fuel atomization, combustion and formation of pollutant emissions in diesel engine. To improve the process of fuel injection, CFD packages are used. Since CPU times are often high, partial models are used for the analysis. In presented paper the influence of different density of mesh on the cavitation phenomena is being analysed. The theoretical backgrounds of the cavitation occurrence presented in the first part of the paper are followed by the numerical analyses of two-phase flow in same simplified nozzle models. The numerical analyses are made using computation fluid dynamic (CFD) program Fire. The numerical analysis is made for two different types of fluid, diesel (D2) and biodiesel (B100). Numerical analysis also includes various densities of meshes and their influence on results. The two-phase flow is analysed using a two-equation approach, where all conservation equations are solved for every phase. Numerical analysis results are compared to the experimental observations of the two-phase flow available from the literature. The results are compared for various meshes and various fluid types. The results show that higher pressure yields mode cavitation and point out the importance of mesh densities.

Keywords: *two-phase flow, cavitation, injector nozzle, CFD, nodalisation analysis*

1. Introduction

The fuel injection system of the diesel engine plays the dominant role on the fuel spray formation which affects the combustion and pollutant formation processes. It is well known that the injector nozzle flow have a significant role on spray, but it is still not very well discovered. The major problem represents the dimension of the nozzle channel flow areas. The measurements are more or less limited to the measurements of the nozzle flow coefficients at the steady state conditions.

At the other side computation fluid dynamics (CFD) packages become on importance in recent years. The main problem that occurs at the CFD analyses is the required computational time. To

shorten the time needed for calculation of the single analysis, simplified model are used (1/2, 1/4 etc.). Since the typical diameter of the diesel injection nozzle is usually less than 1 mm and the pressure differences could even exceed the 200 MPa, the experimental analyses are quite difficult and limited. Upon the above-mentioned problems the cavitation in the diesel injection nozzles should be analysed mode carefully. The cavitation phenomena should be taken into the consideration when designing or improving the injection nozzle and injection systems.

2. Theoretical backgrounds

Cavitation bubbles form because of low static pressure that occurs near a sharp inlet corner in the nozzle flow. If the corner of the inlet is sufficiently sharp, the flow tends to separate and form a contraction inside the nozzle, which reduces the area through which the liquid flows. This reduced area is accompanied by increase in velocity, as predicted by conservation of mass. Conservation of momentum predicts that the acceleration of the liquid through the vena contracta causes a pressure depression in the throat of the nozzle. The low pressure inside the throat of the nozzle may fall below the vapour pressure of the liquid, causing cavitation. A simple sketch of this flow is presented in Fig. 1. Cavitation flow does not, however, strictly adhere to this simple idealization. The formation of the bubbles is sensitive to the geometry of corner and any imperfections in the nozzle shape. The cavitation is also very sensitive for the quality of the liquid. Furthermore, cavitation inception may occur at pressure below the vapour pressure. Another complication is that the cavitating flow is transient and fully three-dimensional. The location of the vapour is not steady and it is usually also not symmetrical.

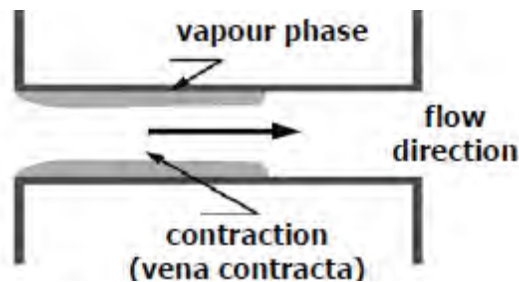


Fig. 1. Simplified sketch of the cavitation in the nozzle

From the experiments made by other authors several types of cavitation in the injection nozzle could be observed. In the nozzle hole fluid starts to evaporate at the sharp inlet edge.

3. Analysis

3.1. Numerical models

Numerical analyses were done by using the CFD program FIRE.

3.2. Computation model

To analyse the flow characteristics of the in-nozzle flow different nozzle models were made. Since some analysis shown, that the pressure drop in nozzle is significant only in the area of the needle seat, sac chamber and nozzle holes, the meshes were modelled only for the above mentioned parts. For the maximal needle lift of 0.3, two different nozzle models, representing real size and one half of the nozzle were made. Since first analyses show no significant changes between the results of the real size and one half model for further research we used one half model.

3.3. Initial and boundary conditions

According to steady state analysing conditions, pressure boundary conditions at the in- and outlet are specified. The fluid used for analysis are the diesel D2 and biodiesel B100, with the temperature of 293.15 K, the density 825 kg/m^3 for D2 and 875 kg/m^3 for B100 and dynamic viscosity of 0.00245 Ns/m^2 for D2 and 0.00265 Ns/m^2 for B100. K- ϵ model is employed. Since maximal velocities of used fuel are much smaller than the speed of sound, the fluid is assumed to be incompressible.

3.4. Density of used meshes

Numerical analysis includes various densities of meshes. Densities for used meshes are presented in Tab. 2 and on Fig. 2. All used meshes are block-structured. Standard approach is to use block-structured types of meshes, while we had a chance to compare (time) it with structured, to study the differences between them.

Tab. 2. Various densities of meshes

Name	Number of used elements	Type
mesh 1	22.000	block-structured
mesh 2	41.280	block-structured
mesh 3	178.880	block-structured
mesh 4	1.202.192	block-structured
mesh 5	2.479.616	block-structured
mash 6	22.100	structured
mesh 7	41.480	structured
mesh 8	179.580	structured
mesh 9	1.204.092	structured
mesh 10	2.499.216	structured

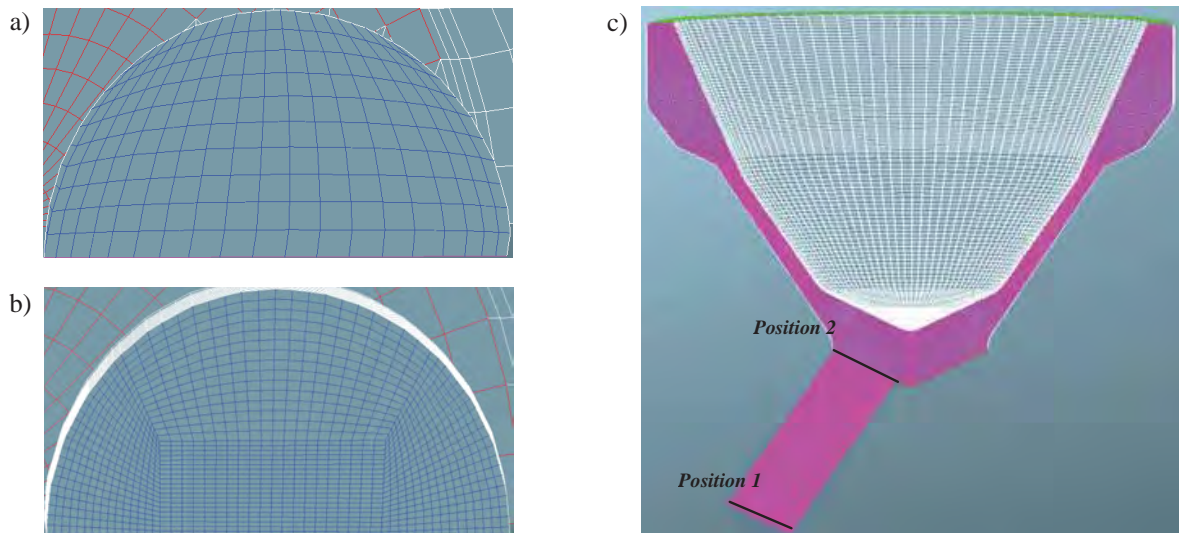


Fig. 2. Structured (a) and block- structured (b) types of used meshes, positions of point were the result are taken (c)

4. Results

Results were taken on two different positions on the injector hole. The positions are presented in the Fig. 2c. All the results were taken at the same time (0.001765s). It is assumed that the needle is fully opened at this time.

4.1. Results of the numerical analysis

Velocity profiles and volume fraction distributions derived from the CFD analyses for differently meshed models are presented in following figures. Figure 3 (left: back view; right: front view) show the results of volume fraction distribution in nozzle hole with various mesh densities.

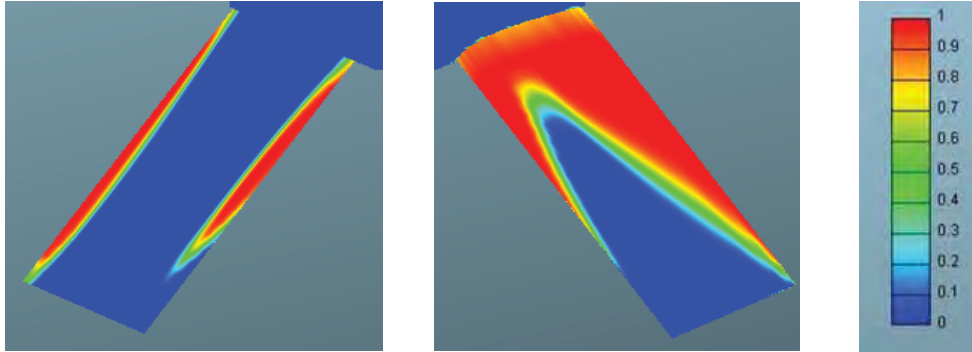


Fig. 3. Volume fraction (mesh 1)

The velocity profiles and volume fraction distributions in nozzle holes with various densities, Fig. 3 are almost identical for all used meshes. The results indicate already known fact that the outflow velocity is higher at holes with smaller inclination angles, what results in higher flow coefficient at those holes.

The results of the numerical analysis for velocity profiles and volume fraction on block-structured meshes of various densities are presented in Fig. 4-6, while Fig. 7 shows the results for structured mesh.

The velocity profiles and volume fraction distribution at the nozzle inlet are almost identical (Fig. 4 and 6). Comparison of the results in Fig. 6 and 7 show how different type significantly influences the velocity profiles. When we use structured (Fig. 7) mesh the velocity profiles are rougher than in case of block-structure (Fig. 6). When we compare the results for various density meshes there are difference between meshes with higher densities compared to meshes with lower density (Fig. 4 and 6). The vapour volume fraction results are presented on Fig. 4. Results for mesh 1 shows the numerical (interpolation) caused by interpolation on insufficient number of elements (peak at length 0.0001 m). Peak at length 0.00055 m for meshes 4 and 5 are caused by vapour cloud which is spreading through the hole till it reaches the outlet.

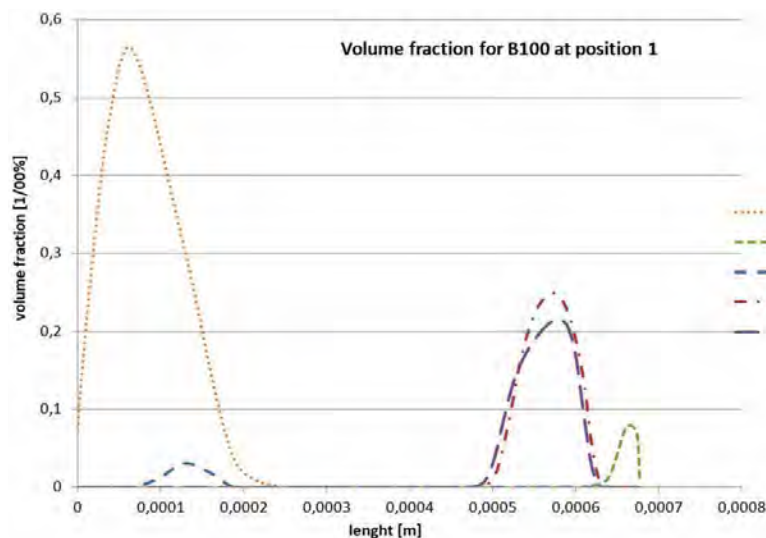


Fig. 4. Volume fraction for block-structured type and various densities of used meshes at position 1

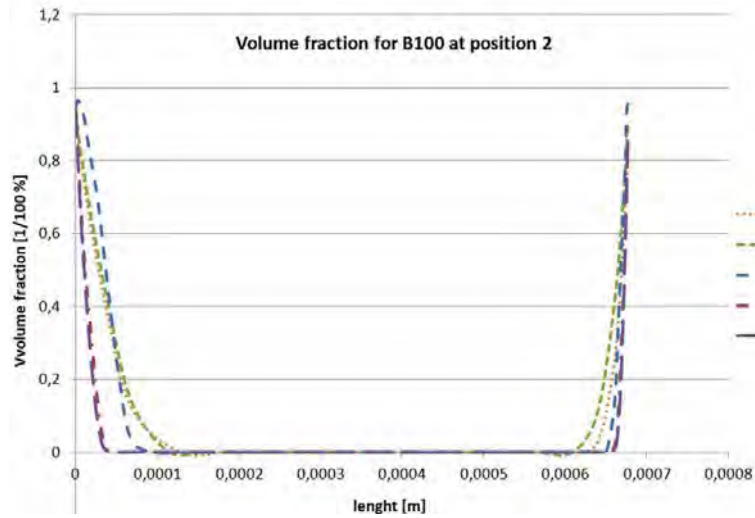


Fig. 5. Volume fraction for block-structured mesh and various densities of used meshes at position 2

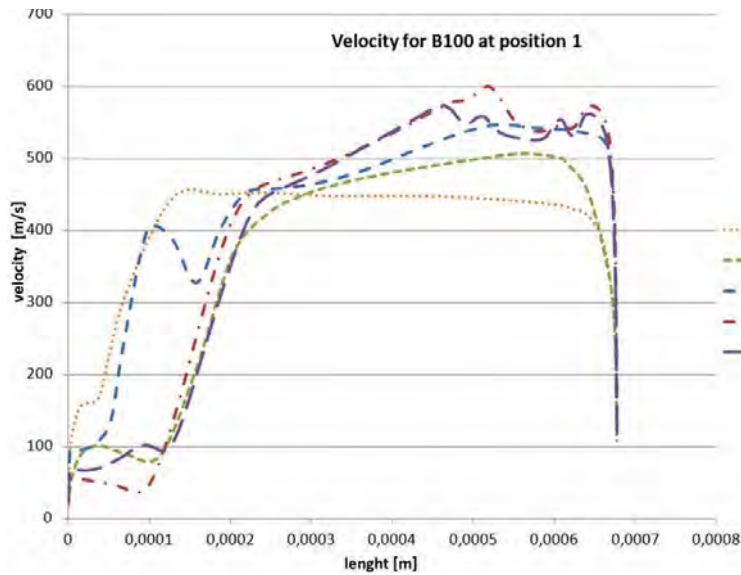


Fig. 6. Velocity profiles for block-structured mesh and various densities of used meshes at position 1

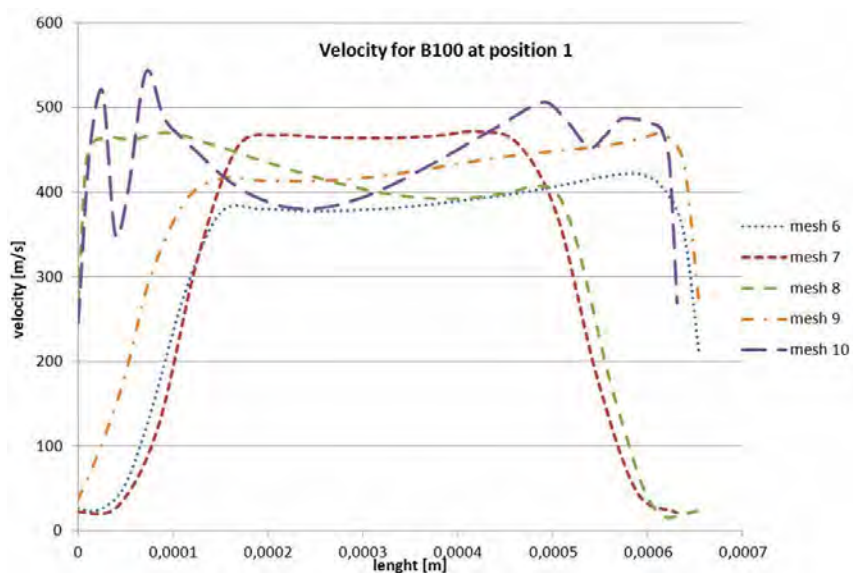


Fig. 7. Velocity profiles for structured mesh and various densities of used meshes at position 1

As already stated the results of the analysis on the real size and one-half model of the nozzle showed no significant difference. Velocity fields, velocity profiles on the outlet and pressure distributions are comparable. This fact is shown on Fig. 8.

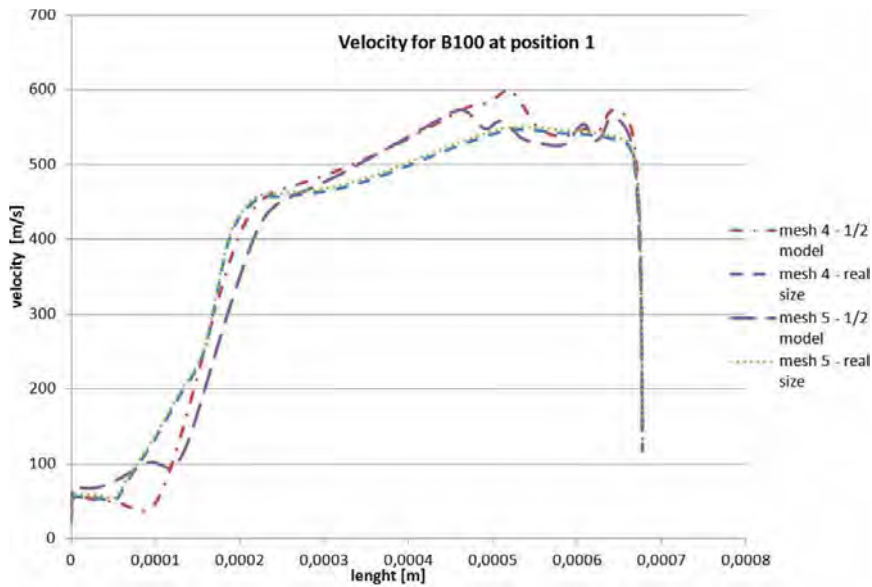


Fig. 8. Velocity profiles for block-structured type and used models at position 1

The difference in velocity in case of mesh 4 – ½ model and mesh 4 – real size model is maximum 2 % (Fig. 8). The calculated value of the flow coefficient are 0.688 for real size and 0.689 for half model, where the CPU times for one half model are only 43.5 % of the CPU needed for the real size model calculation.

The results of influence of different type of fluid (D2 and B100) in nozzle showed no significant difference (Fig. 9). Velocity field, velocity profiles and volume fraction distributions on the outlet are comparable. The results are comparable because the density of used fluids is similar and not so much different (difference is about 50 kg/m^3). The difference between dynamic viscosities is also very small and has no effect on the results. Values for mass-flow and viscosity were different (taken from experiment), while pressure and temperature were the same for both fuels.

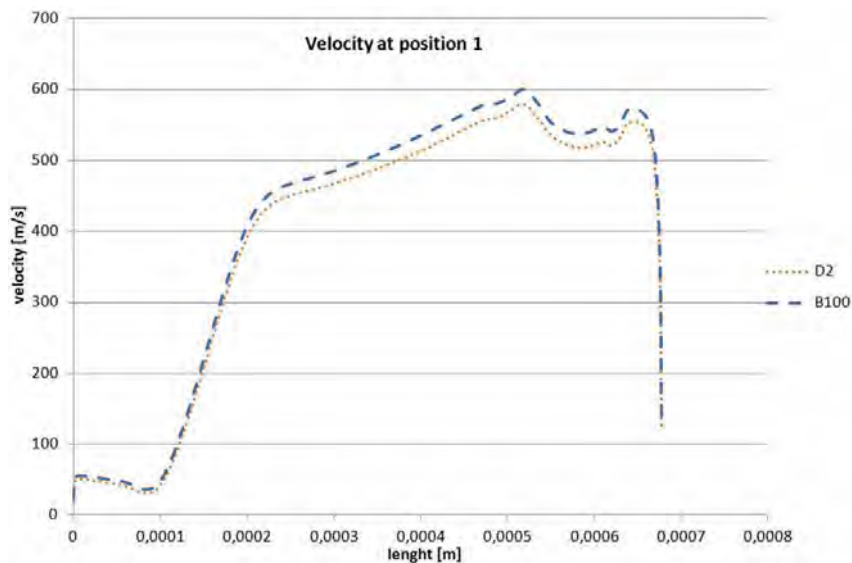


Fig. 9. Velocity profile for different type of fluid (D2 and B100) at position 1 (mesh 4)

The pressure distributions presented in Fig. 10. show the very low pressure area in the recirculation zone. This means, that the velocity at the outlet is higher and it is also more uniform. Higher output velocities result also in better atomisation of injected fuel.

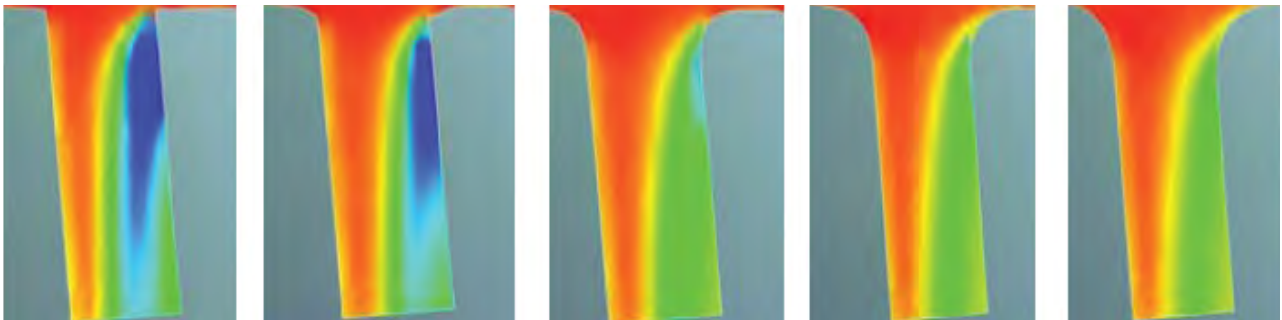


Fig. 10. Pressure distribution in the nozzle hole (central cut, from left: sharp edge, 0.05, 0.1, 0.15 and 0.2 mm, Scaling: dark colours-low pressure regions, light colours –high pressure)

5. Conclusion

Considering above-mentioned results the following conclusions could be made:

- the numerical analysis shows that higher pressure differences yield more cavitation,
- the results of the numerical analysis are comparable to the experimental results,
- the results of the numerical analysis show that the block-structured type of mesh is better for numerical analysis in nozzle holes,
- it is recommended to use meshes with higher density (higher than 1 mil elements) in case of in-nozzle flows, but meshes with lower density (between 100,000 and 500,000) can also be used for fast estimations,
- the results of numerical analysis for different types of fluids (D2 and B100) are comparable, however we recommend more simulations with additional parameters (pressure, temperature...) to be made.

References

- [1] Arcoumanis, C., et al., *Analysis of the Flow in the Nozzle of a Vertical Multi Hole Diesel Engine Injector*, SAE Paper 980811.
- [2] FIRE Version 9 – *Users manual, Multiphase flow*, AVL, 2009.
- [3] Ganippa, L. C., et al., *The structure of cavitation and its effect on the spray pattern in a single hole diesel nozzle*, SAE paper 2001-01-2008.
- [4] Ganippa, L. C., et al., *Comparison of cavitation phenomena in transparent scaled-up single hole diesel nozzles*, 4th international symposium on cavitation, California Institute of Technology, Pasadena, 2001.
- [5] Göschel, B., *Einspritzdüsenverschleiß*, Forschungvereinigung Verbrennungskraftmaschinen e.V., Heft R245, 1974.
- [6] Hardenberg, H., *Die Nadelhubabhaengigkeit der Durchflussbeiwerte von Lochduessen fuer Direkteinspritzdieselmotoren*, MTZ 46, pp. 143-146, 1985.
- [7] ISO-4113:1988, *Road Vehicles- Calibration Fluid Diesel Injection Equipment*.
- [8] Kato, M., et al., *Flow Analysis in Nozzle Hole in Consideration of Cavitation*, SAE 970052
- [9] *Kavitation, Abschlußbericht über die Ergebnisse des Schwerpunktprogramm 1966-1972*, Deutsche Forschungsgemeinschafts, Boppard 1974.
- [10] König, G., et al., *Analysis of flow and cavitation Phenomena in diesel injection nozzles and its effects on spray and mixture formation*, 5 Internationales Symposium für Verbrennungsdagnostik der AVL Deutschland, Baden Baden 2002.

- [11] Melcher, K., Chomiak, J., *Experimentelle Untersuchung der Stroemung durch Dieseleinspritzduesen in stationaer betriebenen Grossmodel*, Bosch Techn. Berichte Nr. 5, 1976.
- [12] Oishi, Y., et al., *A computational Study into the Effect of the Injection Nozzle Inclintion Angle on the Flow Characteristics in Nozzle Holes*, SAE Paper 920580.
- [13] Yoda, T., Tsuda, T., *Influence of injection Nozzle improvement on DI Diesel Engine*, SAE Paper 970356.